Strathprints Institutional Repository

Rosochowska, Malgorzata and Balendra, Rajendram and Rosochowski, Andrzej and Olejnik, Lech (2012) *FE modelling of multi-stage drawing of a miniature tubular component with a middle flange and rounded edge.* In: IDDRG 2012. International Deep-Drawing Research Group, Mumbai, pp. 128-133.

FE modelling of multi-stage deep drawing of a miniature tubular component with a middle flange and rounded edge

Malgorzata Rosochowska[1,a], Raj Balendra[2,b], Andrzej Rosochowski[2,c] and Lech Olejnik[3,d]

[1]Advanced Forming Research Centre, University of Strathclyde, Glasgow, UK

[2]Design, Manufacture and Engineering Management, University of Strathclyde, Glasgow, UK

[3]Institute of Manufacturing Technologies, Warsaw University of Technology, Warsaw, Poland

[a]m.rosochowska@strath.ac.uk, [b]raj.balendra@strath.ac.uk, [c]a.rosochowski@strath.ac.uk, [d]l.olejnik@wip.pw.edu.pl

Keywords: deep drawing, sheet metal forming, FE simulation, anisotropy

Abstract. An FE model to simulate a multi stage deep drawing/redrawing process of a miniature tube with a flange mid-way its length was developed. At the final stage of the manufacturing sequence the upper trimmed end of the tube was curled. The curling operation progressed to a point until the curling force caused the collapse of the flange. The objective of the research was to define the limiting geometric and process parameters for the curling operation. Two alternative FE models, 3D and 2D axisymmetric, were adopted. The 3D model was used to investigate anisotropic behaviour of the material during the first drawing operation in order to establish whether there were conditions that warranted resorting to 3D modelling. Eventually the 2D axisymmetric model was used to simulate the material flow in all 11 forming operations. Additionally a springback analysis was performed after each operation; strain and stress states resulting from the unloading of the formed component were used as the initial states for the succeeding operations. The FE simulation results matched the experimental results and led to the conclusion that the accuracy of trimming played a critical role in the outcome of the curling operation.

Introduction

The object of investigation was a multi stage deep drawing/redrawing process of a miniature tubular component (Fig. 1) with a flange approximately mid-way the length of the tube. At the final stage of the manufacturing sequence, the upper end of the tube was curled. The curling operation progressed to a point until the curling force caused the collapse of the flange. The component was out of production but it was chosen as a test case to check whether FE simulation might be capable of defining the limiting geometric and process parameters for the curling operation, so it could be completed without the collapse of the flange. Development and refinement of the FE model was supported by mechanical testing of the material used.

An alternative approach to simulation of multi-stage deep drawing processes has been reported in [1]. In that approach, the simulation was allowed to start from any intermediate operation, provided there was information on material properties at the end of the preceding operation.

Fig.1 Component.

Tensile testing of the sheet material

Tensile tests were performed to define uniaxial deformation characteristics and the anisotropic behaviour of the 305 stainless steel 0.267 mm thick sheet used for the component. For industrial processing, the material was supplied as a 20 mm wide strip. However, considering testing requirements, the work-material for the mechanical testing was supplied as 800 mm wide sheet. The mechanical properties were assumed to be similar since narrow strips used for manufacturing operations were produced by slitting these wider sheets. Test specimens, 100 mm long, were cut at 0º, 45º and 90º to the rolling direction using wire EDM (Fig. 2). Prior to being subjected to straining, a grid was photo-etched on the surface of each specimen; this enabled both, in-process and post-process measurements of the plastic deformation of the specimens. The Swift expression, $\sigma=A(B+\varepsilon)^n$, was used to define the relationship between true stress and true strain. The Lankford plastic anisotropy factors R were determined using only the data derived over the uniform-deformation range; the original gauge length l_0, width w_0 and the thickness t_0 and the same dimensions measured prior to necking of the specimen were used to calculate values of R. Values of A, B, n and Lankford anisotropy factors R are shown in Table 1; also shown in this table are the 0.2% proof strength, the ultimate tensile strength and elongation at fracture. Each data is the average of values obtained in five tests. R values of the sheet material are different from that reported in [2] for the same type of stainless steel (R_0=0.9, R_{45}=1.26 and R_{90}=0.86).

Fig.2 Packs of tensile test specimens.

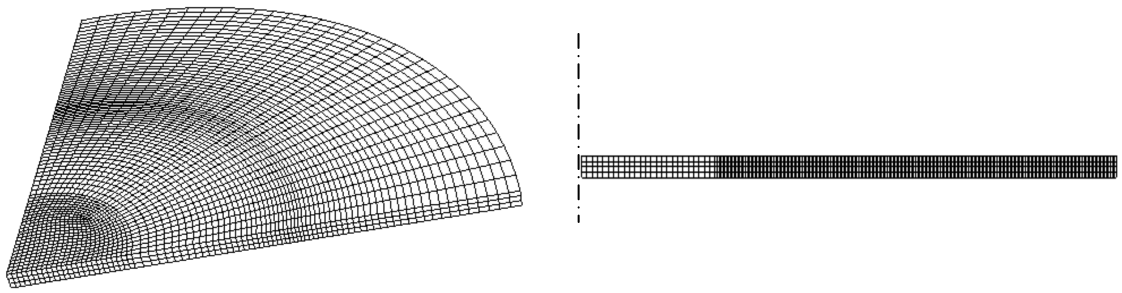
Table 1: Properties of sheet material

Orientation of test specimen	Proof strength	Ultimate tensile strength	Elongation at fracture	Lankford anisotropy factors	Swift's constants		
	[MPa]	[MPa]	[%]	R	B	A	n
0°	233	577	48	0.75	0.041	1233	0.52
90°	221	558	63	1.05	0.034	1313	0.53
45°	222	525	48	1.02	0.027	1203	0.51
				0.955$^{(1)}$			
				-0.12$^{(2)}$			
$^{(1)}$ $\overline{R} = (R_0 + 2R_{45} + R_{90})/4$ – normal anisotropy $^{(2)}$ $\Delta R=(R_0 - 2R_{45} + R_{90})/2$ – planar anisotropy							

Finite element modelling

Finite element simulation of the drawing/redrawing sequence was performed using Abaqus/Explicit. Two alternative FE modelling approaches, 3D and 2D axisymmetric, were adopted. The 3D model was used to simulate only the first drawing operation in order to evaluate the anisotropic behaviour of the sheet material and to enable an understanding of the scale of errors that were likely to occur if the more rapid, 2D simulations were to be adopted to simulate the whole sequence of operations.

In 3D model, one quarter of the blank was modelled using appropriate boundary conditions on each of the two symmetry planes, z-x and z-y. Solid linear hexahedral elements were used to create the mesh; three elements spanned the thickness of the blank. A total of 7392 elements (Fig. 3a) were used to model the blank. The tools were modelled as rigid bodies.



a) b)

Fig.3 Mesh of the blank used in (a) 3D and (b) axisymmetric model.

The 2D axisymmetric model was used to simulate the sequence of 11 successive forming stages; springback analysis was performed after each stage using Abaqus/Standard. This meant that the strain and stress states resulting from unloading were used as the initial states for the following operations. In all operations, the tooling was modelled as a perfectly rigid body. Axisymmetric solid elements were used to model the blank, the thickness of which was modelled using four elements. The length of the elements along the blank was selected depending on the anticipated elongation; 085 mm long elements were used in the area that remained in contact with the face of the punch while the rest of the blank was meshed with 0.03 mm long elements. A total of 948 elements (Fig. 3b) were used to model the blank.

The sheet material was modelled as an elastic plastic material with isotropic elasticity; the Mises isotropic yield criterion was adopted for the 2D axisymmetric model and the Hill's anisotropic yield criterion for the 3D model. The Mises yield criterion used was that derived for the rolling direction. The anisotropic yield ratio parameters R_{ij}, used in the Hill's function, were calculated using R values derived experimentally (Table 1). Table 2 shows material properties used in the FE simulation.

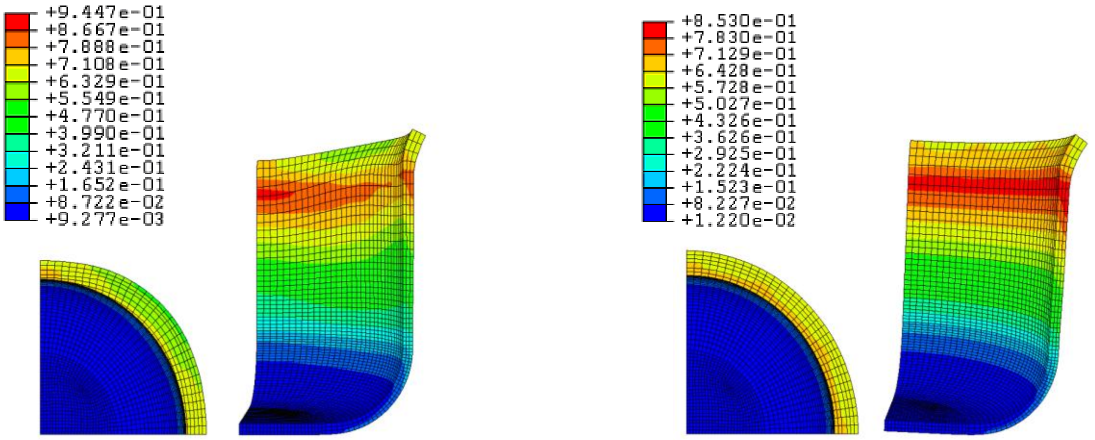
Table 2: Material properties used in FE models

Young's modulus	193000 MPa
Poisson's ratio	0.3
Density	7800 kg m^{-3}
Work hardening	$\sigma=1233(0.042+\varepsilon)^{0.52}$ MPa
Anisotropic yield ratios	$R_{11}=1$, $R_{22}=1.1508$, $R_{33}=1.0023$, $R_{12}=0.9916$, $R_{23}=1$, $R_{13}=1$

Friction conditions were modelled using the Coulomb's law. Results of preliminary simulations suggested that a coefficient of friction equal to 0.1 was good approximation of the conditions that prevailed at the blank/blank holder interface and the blank/die interface. However, if the same friction coefficient was used to simulate friction on the punch surface, the sheet material that was drawn past the punch radius displayed a significant reduction of the wall thickness. Following the common industrial practice of roughening the punch radius using a sand paper with a view to increasing the friction at this point of the tool, the FE model subsequently used employed two different values of friction coefficient, 0.25 on the punch surface and 0.1 everywhere else; this approach resulted in a more exact replication of the drawing process.

Results of simulation for first drawing operation – 3D model

The geometry of the cup and strain distribution in the first operation, computed using a 3D model with an anisotropic and isotropic material definition, is shown in Fig. 4a and 4b, respectively. These results demonstrate that the sheet material is marginally anisotropic. However, as shown in Fig. 5, a



a) b)

Fig.4 Equivalent plastic strain distribution in the component drawn in the first operation for (a) anisotropic material modelled and (b) isotropic material model.

small degree of earing was observed in the industrial process. The simulations showed that the maximum equivalent plastic strain sustained by the sheet material was 0.945 and 0.853 for anisotropic and isotropic material, respectively. However, the band of material with the largest strain was eventually trimmed off the component prior to the curling operation thus not having influence on this operation. Further, strain distribution below this band was similar for both materials. This suggested that the first operation that is critical to the development of the model and consequently the whole operational sequence could be accurately analysed using a 2D axisymmetric model.

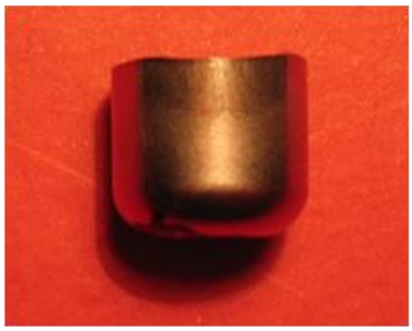


Fig.5 Shape formed in the first drawing operation.

Results of simulations for axisymmetric model

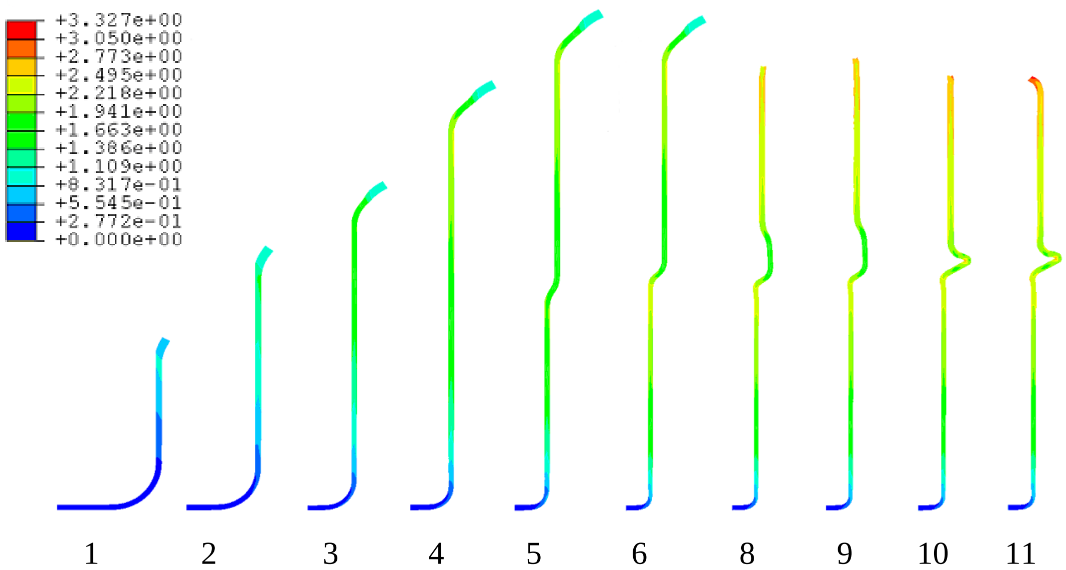
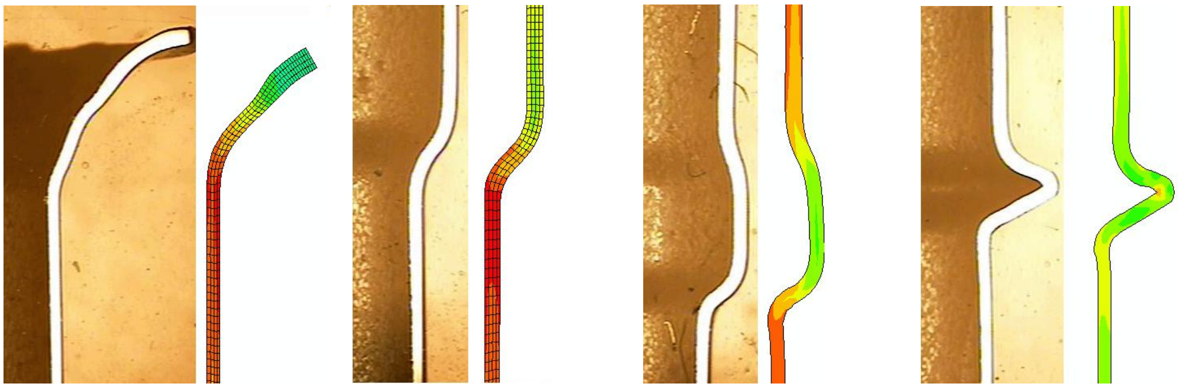


Fig.6 Equivalent plastic strain accumulated in subsequent operations (clipping 7 is not included).

The component shape and equivalent strain evolution in subsequent operations are shown in Fig. 6. During the operation number 7, which is not included in this figure, the flange formed in the preceding operations was trimmed by clipping. The required tube length after clipping was obtained by removing finite elements that formed the flange (∗MODEL CHANGE in Abaqus). Forces, the removed region exerted on the remaining part of the model at the nodes along the clipping line, were gradually ramped down to zero. The computed shapes show good agreement with the formed shapes during each production stage. Fig. 7 shows comparison of some features of the components in the 4^{th}, 6^{th}, 8^{th} and 10^{th} operation obtained in the FE simulation and observed in the real process.



Operation 4 Operation 6 Operation 8 Operation 10

Fig.7 Comparison of formed and computed shapes in the operations 4, 6, 8 and 10.

Curling operation and collapse of the flange

During the last operation the open end of the component was curled inward to a specific diameter. Fig. 8 shows the shape of the component before and after the curling operation. Fig. 9 shows the variation of the opening diameter and the punch force with the displacement of the punch during curling. Up till the punch displacement of approximately 1.5 mm, the component having been positioned on the edge of the die is moving into the die cavity. Forming of the curl commences at point A and is expected to be completed when the opening diameter is in the range 2.06-2.21 mm. These values are marked as “max” and “min” in Fig. 9; punch force corresponding to these opening

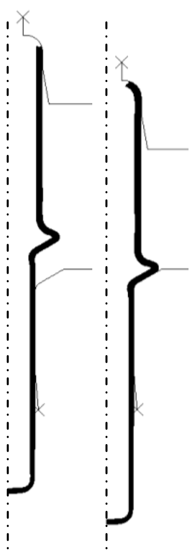


Fig.8 Curling operation before and after.

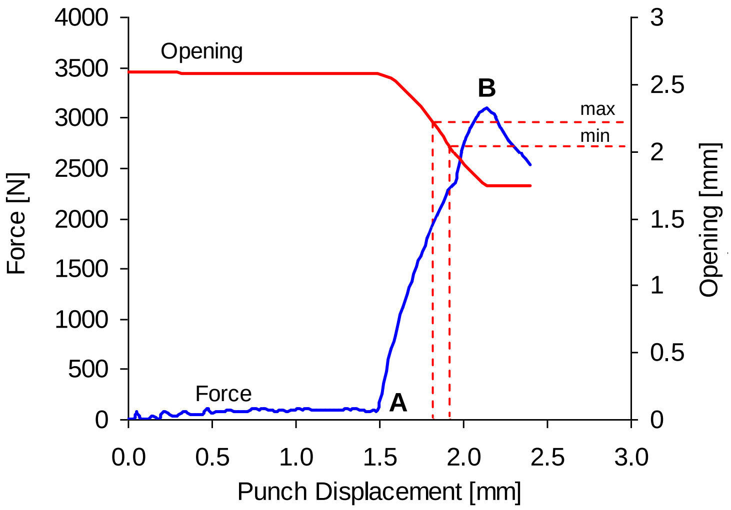
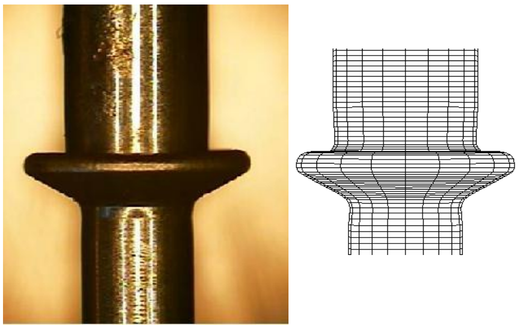


Fig.9 Variation of punch force and opening diameter with punch displacement during the curling operation; max and min refer to tolerances specified.

diameters is 1950 N and 2250 N, respectively. At point B, the movement of the punch results in the collapse of the flange (Fig. 10) so the opening diameter is arrested.



a) b)

Fig.10 Collapsed flange: (a) in the formed component, (b) simulated.

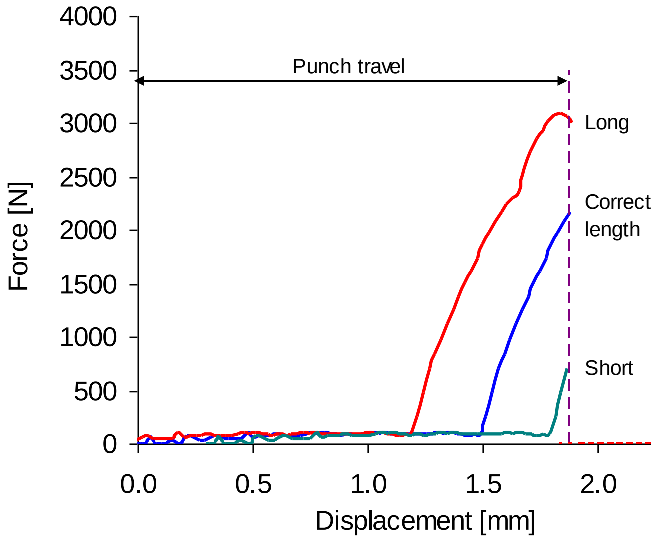


Fig.11 Progress of the curling operation in relation to the trimmed length of tube.

The difference in punch displacement between the minimum and maximum opening diameters is 0.1 mm; further, the difference between the punch position for acquiring the mean opening diameter and the maximum force is 0.25 mm. It follows that the length of the tubular section above the flange is a key parameter in determining the quality of curling. Since punch displacement is constant, a variation in the length of the tubular section leads to three possible scenarios (Fig.11): the opening diameter being within the specified range (correct tube length), initiation of the flange collapse (tube too long) and the opening diameter not fully formed (tube shorter then the minimum specified).

Conclusions

Considering the objective of this study, which was to check whether and how an FE model could be developed to simulate a multi stage industrial sheet metal forming process, the key findings are:

- Given that the sheet material is marginally anisotropic, most results required to draw valid conclusions may be generated using a 2D axisymmetric FE analysis.
- Approach based on the simulation of all operations, with springback taken into account and the stress/strain state transferred between the operations, enables good representation of a multi stage drawing/redrawing process.
- Trimming is critical for the outcome of the curling operation; essentially, this shows that the permissible margin of error in the trimmed length is of an order of ±0.1 mm.

References

[1] J. Post, C. de Vries, J. Huetink, Validation tool for 2D multi-stage metal-forming processes on meta-stable stainless steels, J. Mater. Process. Tech. 209 (2009) 5558-5572.

[2] S. Gallée, P.Y. Manach, S. Thuillier, Mechanical behavior of a metastable austenitic stainless steel under simple and complex loading paths, Mat. Sci. Eng. A 466 (2007) 47–55.